Cesar 3D

Tutorial 3g.01

Excavation works on an embankment

1. PREVIEW

1.1. Tutorial objectives

With help of this tutorial, user will access the main feature of geometry modelling in CESAR 3D. He will also analyse several main issues of geotechnical engineering:

- modelling the construction stages of an embankment,
- analysis of the overall stability with service loading,
- evaluation of the impact of excavation works on the global behaviour.

Note that several hypotheses are made so that the study is fitted for an evaluation purpose (calculation times for example). For real engineering purposes, user must be aware on the importance of choice of materials properties, mesh density and other calculation settings.

1.2. Problem specifications

Geometry

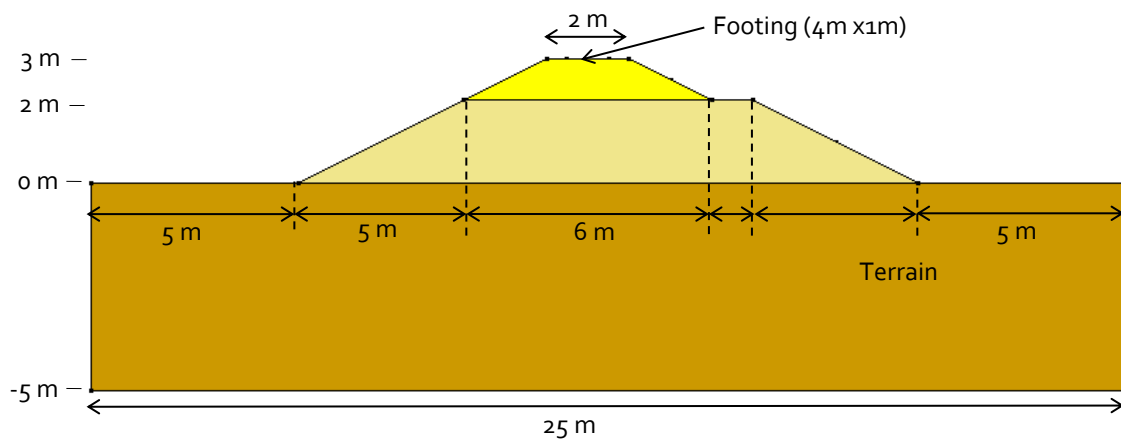We present below a view of the project at final state (2d view).

Figure 1: plane view of the embankment

Analysis stages

1. Initialisation of stresses inside the support terrain.
2. Construction of the lower embankment.
3. Construction of the upper embankment.
4. Application of pressure on the footing.
5. Excavation works (2,5 m depth) in the slope.

Materials

	γ_h (kN/m³)	E (MPa)	ν	c (kPa)	φ (°)	ψ (°)
Terrain	18	25	0,35	10	35	5
Embankment	20	40	0.3	30	25	0

2. GEOMETRY AND MESH

2.1. *General settings*

1. Run CESAR 3D.

<u>Units</u>

1. Set the units in the menu ***Units*** ^(m).
2. On the left, select the **General** section and set the *Length* unit to **m** in the toolbox.
3. On the left, select the **Mechanic** section and set the *Force* unit to **MN**.
4. In the same way, select the *Displacement* unit to **mm**. Adjust the number of digits to 0.0.
5. Finally, select the leaf *Surface* and set the display in scientific digits.
6. Click on ***Apply*** to close.

💡 Use « Save as default" to define this unit system as the default user environment.

2.2. *Geometry*

A new project always starts in the tab *GEOMETRY*.

<u>Working plane settings</u>

1. In ***Working plane***:
    -   Set OXY as coordinates system plane.
    -   Set the visible grid to 1m (dX = dY = 1m)

<u>Drawing of the contours</u>

Choose one of the two methods proposed below:

1. Use *Lines* and draw the contour edges using the grid as support.

or

1. Activate ⟋X . The point definition dialog is displayed.
    -   Define the coordinates of the construction points (see figure 1).

2. Use *Lines*.to link the points previously defined.
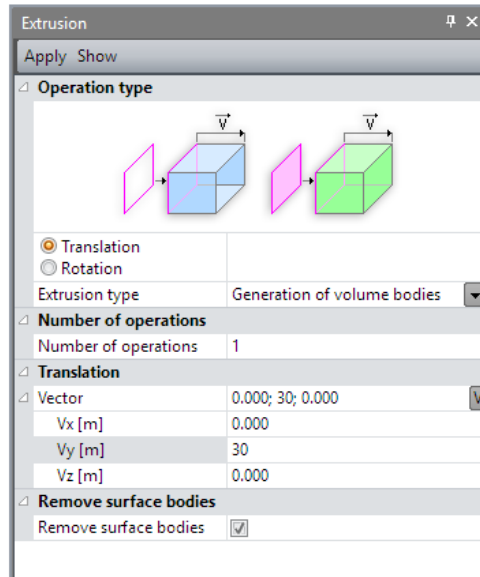
<u>Generation of the surfaces</u>

We can now define the associated surface.

1. Select the previously defined edges.

2. Activate ***Plane Surface***. Keep the default setting "All regions"
3. ***Apply***.

Generation of the volumes

The global volume of the embankment is constructed using an extrusion of the plane surfaces.

1. Activate *Extrusion* and set:
   - *Translation* as operation type,
   - *1* as number of operations,
   - (30 ; 0 ; 0) as vector,
   - Delete the initial surface bodies.

2. Select all the surfaces to be extruded
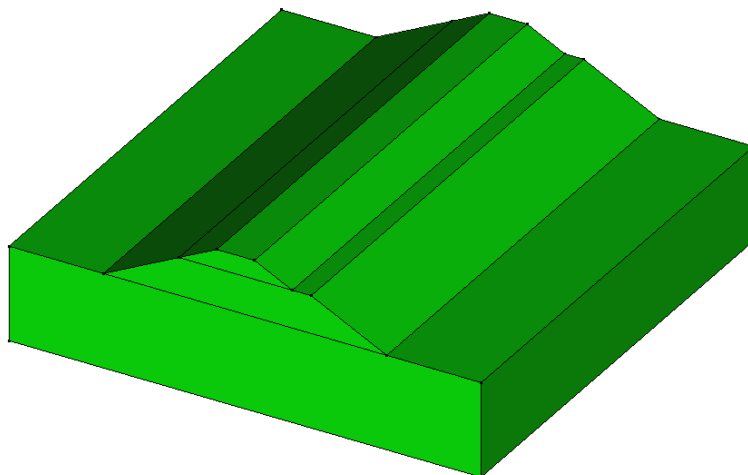   - On the **Selections toolbar**, inactivate bodies selection.
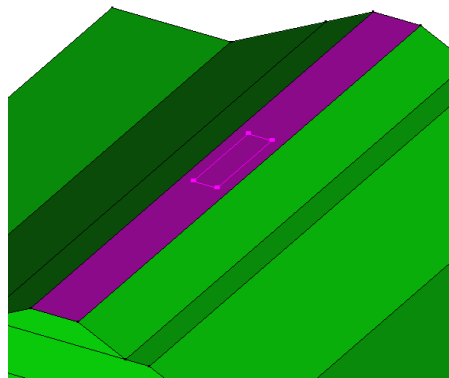

3. *Apply*.

Figure 2: View of the geometry after extrusion

Generation of the footing print:

The footing will be modelled by its print on the embankment. Its Geometry is drawn and « printed » on the upper face of the embankment.

1. Activate ✗ **Points**.
   - Create the points (10 ; 12.5 ; 3), (10 ; 11.5 ; 3), (14 ; 11.5 ; 3) et (14 ; 12.5 ; 3),
   - **Apply**.

2. Use ╱ **Lines**.to link the points previously defined.

3. Activate 🔷 **Surfaces split**.
   - Select the 4 edges created and the top surface of the upper embankment (see figure below).
   - On the **Selections toolbar**, inactivate bodies selection. Activate edges and surfaces selection.

   - **Apply.**

Generation of the surface body support of the footing

The footing is modelled as a shell.

1. Select the edges, limits of the footing.

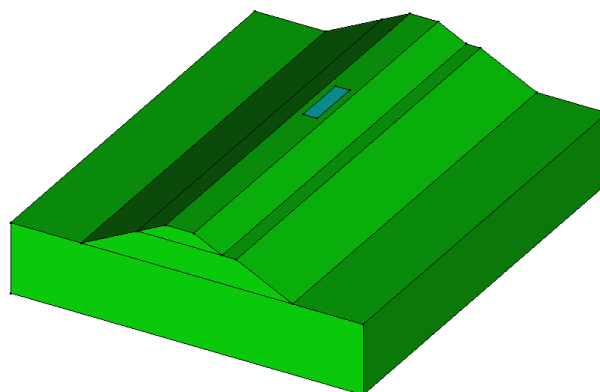2. Activate 🟦 **Surface body on volume**.

3. **Apply**.

4.

Figure 3: View of the spread footing (blue) on the volumes of soil (green)

Generation of the excavated volume

For this step, a box with the dimensions of the excavation is created as tool for the intersection. Then we proceed to the intersection of this box with the existing embankment.

1. Activate **Box**.
   - Choose *Box* defined by « 1 point + Dimensions » as body type.
   - Set the coordinates (12 ; 14 ; 4) for the insertion point of the box,
   - Set (6 ; 4 ; -4) as dimensions of the box.

2. Proceed to the intersection with tool . **Bodies intersection**:
   - Activate *Interactive* and *Cut object + tool*.
   - Click on the volume body of the embankment (body #1) then on the body of the box (body #2).
   - Intersection is completed.
   - Delete the upper volume of the box (result of the intersection) that will not be useful in the model.
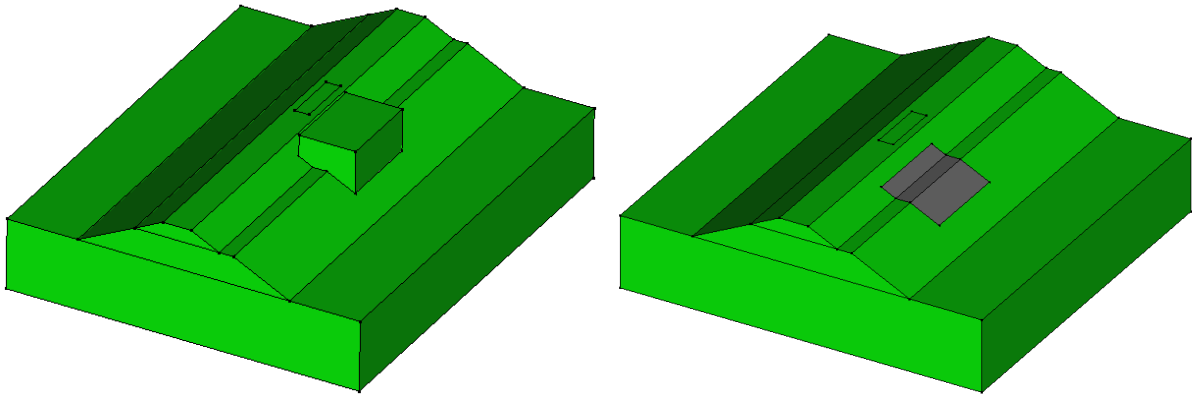

Figure 4: View of the model before and after the intersection

Dissociation of bodies:

During extrusion operation, the initial selection being a group of 3 surfaces, the result is one single body made of the association of 3 volume bodies. In order to distinguish support soil from lower and upper embankment, it is necessary to split them.

1. Activate **Explode bodies**.
2. Select the volume body.
3. **Apply**.

2.3. Mesh

<u>Mesh density</u>

💡 In the scope of this tutorial, the mesh will be loose in order to reduce computation times.

1. Go to the tab *MESH* on the project flow bar to start the definition of divisions along lines.
2. Select all the edges of the model.
3. Click on ⌷ *Fixed length density* to divide these segments with a fixed length.
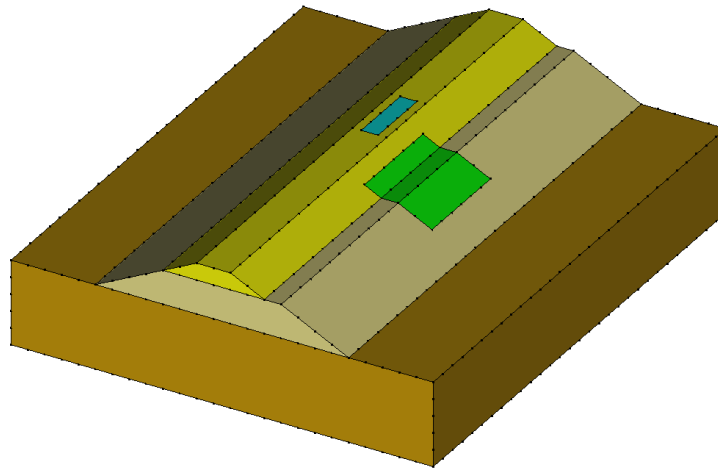   - Set it to **1m**.
   - *Apply*.

Figure 5 : view of the density before meshing

<u>Volume meshing:</u>

1. Select all the volume bodies.
2. Open the tool *Volume meshing*
   - Select "Linear interpolation" as **Interpolation type**.
   - Set "Tetrahedron mesh" as **Mesh type**.
   - Select the default Tetmesh generator and set the **Density factor** to 0,9.
   - Mesh creating function is **Cubic**.
   - *Apply*.
3. Mesh is completed.

<u>Surface meshing:</u>

1. Select the surface body or the footing.
2. Open the tool *Surface meshing*
   - Select "Linear interpolation" as **Interpolation type**.
   - Set "Triangle" as type of mesh.
   - Mesh creating function is **Cubic**.
   - *Apply*.

💡 CESAR-LCPC proposes 3 levels for the meshing procedure of external surfaces of the volume. It enables to definition the progression of the meshing algorithm from small to large elements. The choice is made in Preferences> Mesh > Mesh creating function (Linear = coarse, cubic = dense).
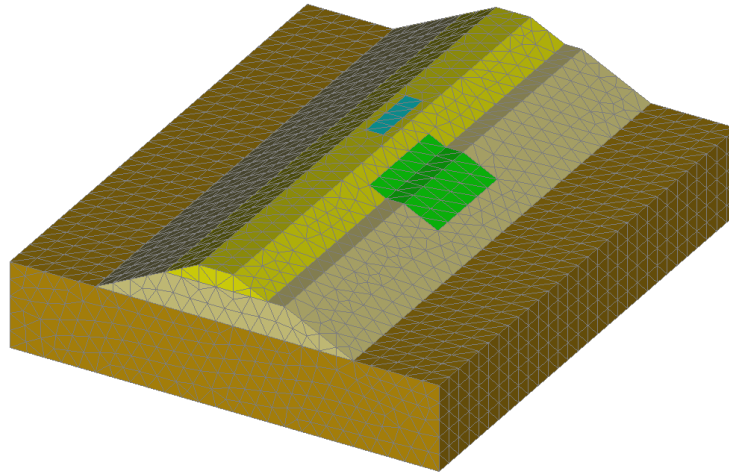
Figure 6: View of the mesh

Bodies definition:

This step is optional but it facilitates the properties and loading assignments for a group of elements during multiple construction stages.

1.   On the **Selection Toolbar**, activate only the option **Select bodies**.

2.   Activate (i) ***Body properties***.

3.   Right-click on the body corresponding to the initial terrain. Enter *Terrain* as name. Specify a colour. Click on **Apply**.

4.   Repeat previous operation for the bodies edition of the base and top embankment, the footing and the excavation.

3. CALCULATION SETTINGS

For modelling the sequence of works stages, it will be convenient to affect all the properties on the complete model and then activate/deactivate bodies in function of their use in the works process.

3.1. Calculation properties for the initial stress field

The initial stress field is initialized with the "Self-weight procedure".

Model definition:

1. On the right side of the working window, the "Study tree view" displays the list of physical domains. Right click on **STATICS**. Click on *Add a model*. A new toolbox is open for definition of the Model.

   - Enter **Initial stress field** as "Model name".
   - Select **MCNL** as "Solver".
   - Tick **Staged construction**.
   - Tick **General initial stress field** as initialization type.
   - Click on *Validate*.
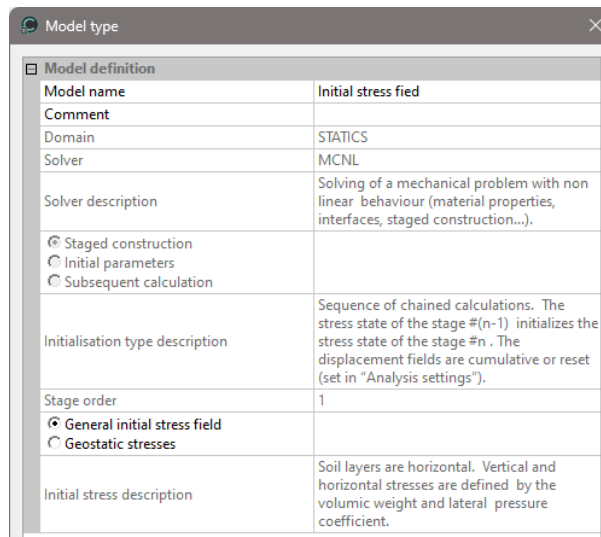
The date tree is now as illustrated below.
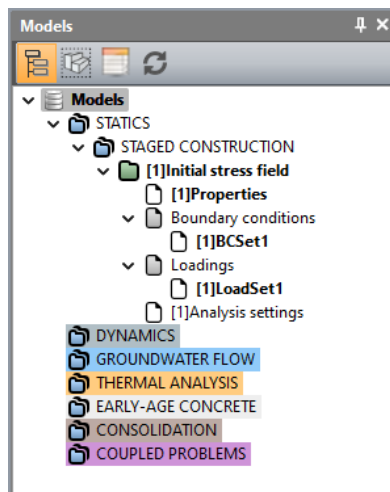
Material properties for the volume bodies:

We define the material library of the study.

1. Click on ⬆️ **Properties for volume bodies**.
2. Give a name for the properties set name (*Embankment* for example).
   - In **Elasticity parameters**, choose "Isotropic linear elasticity" and define ρ, E and ν.
   - In **Plasticity parameters**, choose "Mohr-Coulomb without hardening" and define c, φ and ψ.
3. Repeat previous operation for the *Terrain* properties.
4. Click on **Validate** and **Close**.

	ρ (kg/m³)	E (MN/m²)	ν	c (MN/m²)	φ (°)	ψ (°)
Embankment	2000	40	0,35	0,030	25	0
Terrain	1800	25	0,3	0,010	35	5

Material properties for the footing:

1. Click on 🔷 **Properties for surface bodies**.
2. Give a name for the properties set name ("Concrete" for example).
3. Set the elasticity parameters and the thickness.
4. Click on **Validate** and **Close**.

	ρ (kg/m3)	E (MN/m²)	ν	Thickness (m)
Concrete	2500	35000	0,25	0,15

Assignment of properties sets:

As data sets are created, we affect them to the bodies of the model.

1. Activate 🧱 **Assign properties** tool.
2. Click on ⬆️ **Properties for volume bodies**.
3. Select the volume body corresponding to the embankment.
   - Select the *Embankment* properties set in the list.
   - **Apply**.
4. Select the volume body corresponding to the terrain.
   - Select the *Terrain* properties set in the list.
   - **Apply**.
5. On the left side, click on 🔷 **Properties for surface bodies**.
6. Select the volume body corresponding to the emfootingbankment.
   - Select the *Footing* properties set in the list.
   - **Apply**.

Boundary conditions:

In a standard way, horizontal displacements are blocked on the vertical limits of the model and total displacements are blocked at the base of the model

1. Activate the *BOUNDARY CONDITIONS* tab.

2. On the toolbar, activate ⊞ to define side and bottom supports.

3. *Apply*. Supports are automatically affected to the limits of the mesh.

💡 Optional. It is possible to modify the default name assigned to the boundary condition, BCSet1. Press [F2]; enter **Standard fixities** for example.
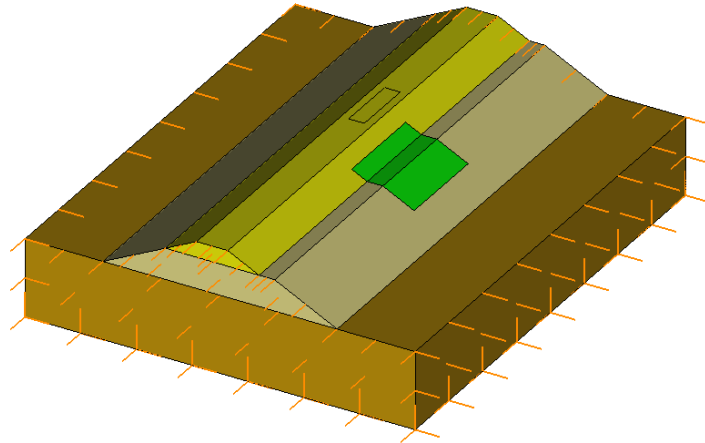

Figure 7 : View of the boundary conditions applied on the model limits

Active/inactive bodies

In this stage, we consider that only the terrain is activated.

1. Activate the *PROPERTIES* tab.

2. Select the volume body representing the Base and top embankment, and the *Footing*.

3. Click on ▯▶▮ *Activate/deactivate bodies*.

4. In the grid, tick "Inactive".

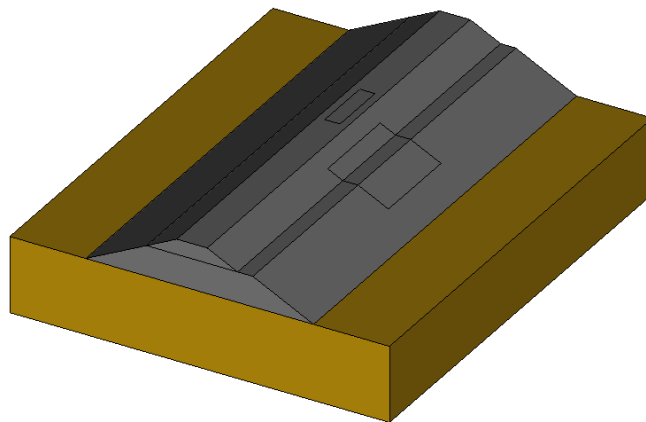5. *Apply*. The volume body is now marked with the neutral colour.

Figure 8 : View of the model after deactivation of bodies

Loading set:

1.   Go to the tab *LOADS*.

2.   On the toolbar, activate ***Gravity forces*** .
- Select all the bodies.
- Click on ***Apply***.

💡 Optional. It is possible to modify the default name assigned to the loading set. Press [F2]; enter **Self-weight** for example.

💡 As a control of the affectation of the gravity forces on 1 or several bodies, the text « Gravity forces » is displayed on the bottom/left corner of the working area.

Calculation parameters:

1.   In *ANALYSIS*, activate **Analysis settings** .

2.   In the **General parameter** section, enter the following values:
- Iteration process:

    Max number of increments: 1

    Max number of iterations per increment: 500

    Tolerance: 0,01
- Method of resolution: 1- initial stresses
- Solver type: Multi frontal

3.   Click on **Validate**.

💡 The option "Calculation with secondary storage" is required when the matrix size of the model will be larger than the random memory (RAM) of the computer.

3.2. First embankment construction

<u>Model definition:</u>

1. In the data tree, right click on the previous model title (**Initial stress field**). In the displayed list, select **Copy of the model**.
2. A toolbox is displayed for definition of this new model.
   - Change the name of the model: **Lower embankment construction**.
   - Select "Stage construction" as initialization type.
   - *Validate* to close the toolbox for model definition.
3. A toolbox is displayed for setting the sharing options of the entities of the model.
   - Copy "Properties",
   - Share "Boundary conditions",
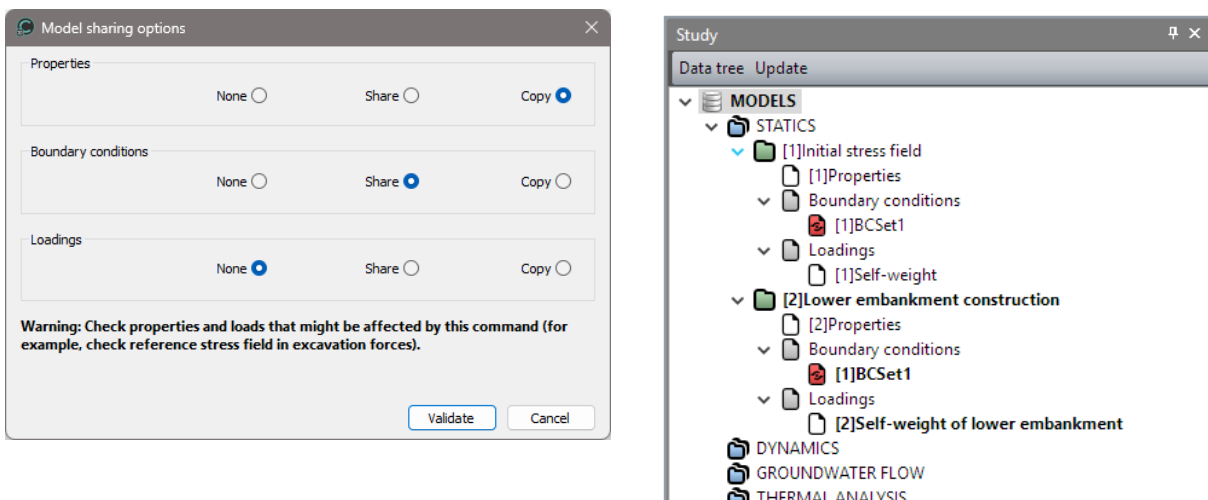   - Do not copy "Loadings".
   - *Validate*

Figure 9 : View of the STUDY tree

<u>Active/inactive bodies</u>

In this stage, we activate the bodies modelling the lower embankment.

1. Select the volume body representing the lower embankment.
2. Click on [icon] *Activate/deactivate bodies*.
3. In the grid, tick "Active".
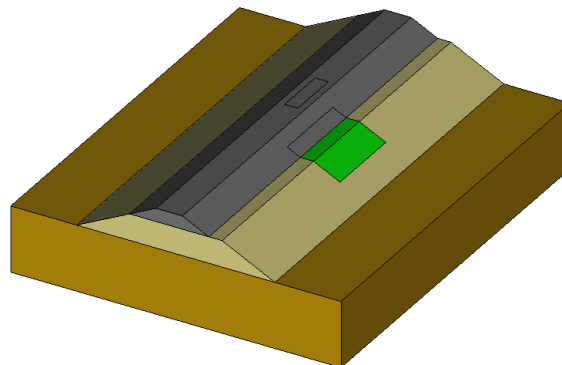4. *Apply*. The volume body is now marked with the initial colour.

Figure 10 : View of the model after activation of lower embankment

Boundary conditions:

There is no change in the boundary conditions.

Loading:

Construction of the embankment body is modelled by activating the self-weight of the added soil mass. We must delete the copied one from **Initial stress field** model, useless here.

1. Go to the tab *LOADS*.

2. Activate   *Gravity forces*.
   - Select the volume bodies of the lower embankment.
   - *Apply*.

Optional. It is possible to modify the default name assigned to the loading set. Press [F2]; enter **Self-weight of lower embankment** for example.

Calculation parameters:

No modifications from previous model.

3.3. *End of the embankment construction*

<u>Model definition:</u>

1. In the data tree, right click on the previous model title (**Lower embankment construction**). In the displayed list, select **Copy of the model**.
2. A toolbox is displayed for definition of this new model.
   - Change the name of the model: **Upper embankment construction**.
   - Select "Stage construction" as initialization type.
   - *Validate* to close the toolbox for model definition.
3. A toolbox is displayed for setting the sharing options of the entities of the model.
   - Copy "Properties",
   - Share "Boundary conditions",
   - Do not copy "Loadings".
   - *Validate*.

<u>Active/inactive bodies</u>

1. Select the volume body representing the upper embankment.
2. Click on   **Activate/deactivate bodies**.
3. In the grid, tick "Active".
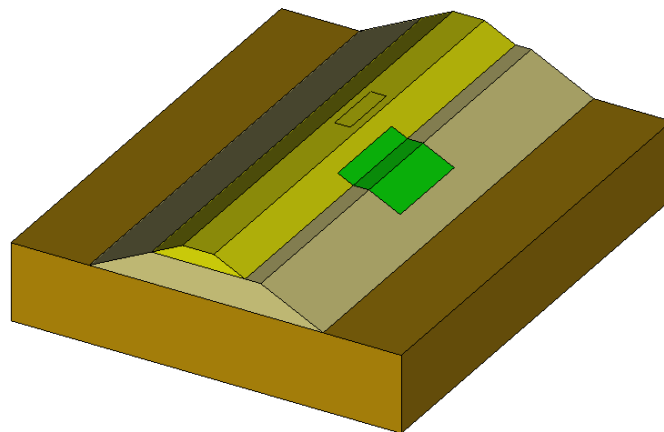4. *Apply*. The volume body is now marked with the initial colour.

Figure 11 : View of the model after activation of upper embankment

<u>Boundary conditions:</u>

There is no change in the boundary conditions.

<u>Loading:</u>

Construction of the embankment body is modelled by activating the self-weight of the added soil mass. We must delete the copied one from previous model, useless here.

1. Go to the tab *LOADS*.
2. Activate   *Gravity forces*.
   - Select the volume bodies of the upper embankment.
   - *Apply*.

<u>Calculation parameters:</u>

No modifications from previous model.

3.4. Permanent loading on the footing

<u>Model definition:</u>

1. In the data tree, right click on the previous model title (**Upper embankment construction**). In the displayed list, select **Copy of the model**.
2. A toolbox is displayed for definition of this new model.
   - Change the name of the model: **Permanent loading**.
   - Select "Stage construction" as initialization type.
   - *Validate* to close the toolbox for model definition.
3. A toolbox is displayed for setting the sharing options of the entities of the model.
   - Copy "Properties",
   - Share "Boundary conditions",
   - Do not copy "Loadings".
   - *Validate*.

<u>Active/inactive bodies</u>

In this stage, we activate the surface body modelling the footing.

1. Select the surface body of the footing.
2. Click on ▯▶▮ *Activate/deactivate bodies*.
3. In the grid, tick "Active".
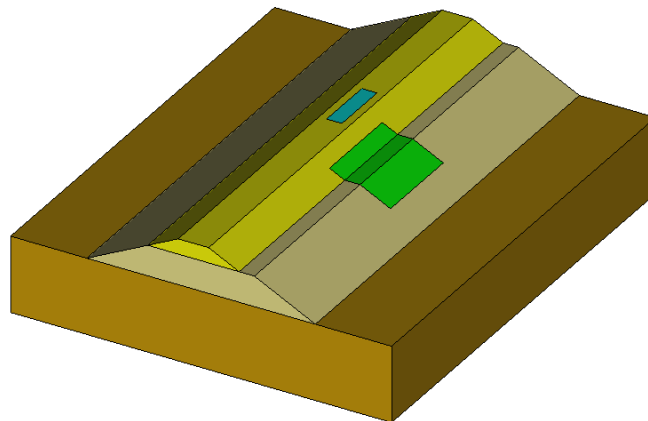4. *Apply*. The surface body is now marked with the initial colour.

Figure 12 : View of the model after activation of the footing

<u>Boundary conditions:</u>

There is no change in the boundary conditions.

Loading:

1. In the tree, rename the default loading set (*LoadSet1*) using [F2] into **Uniform pressure**.
2. Go to the tab *LOADS*.
3. Activate *Surface forces*.
   - Set the values to Py = 100 kN/m² and Pz = -200 kN/m²
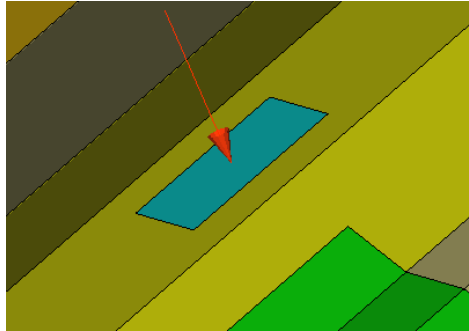   - Apply this pressure by direct click on the face of the *Footing* body.

Figure 13 : Zoom on the loaded footing

Calculation parameters:

No modifications from previous model.

3.5. Excavation

Model definition:

1. In the data tree, right click on the previous model title (**Permanent loading**). In the displayed list, select **Copy of the model**.

2. A toolbox is displayed for definition of this new model.
   - Change the model name: **Excavation**.
   - Select "Stage construction" as initialization type.
   - *Validate* to close the toolbox for model definition.

3. A toolbox is displayed for setting the sharing options of the entities of the model.
   - Copy "Properties",
   - Share "Boundary conditions",
   - Do not copy "Loadings".
   - *Validate*.
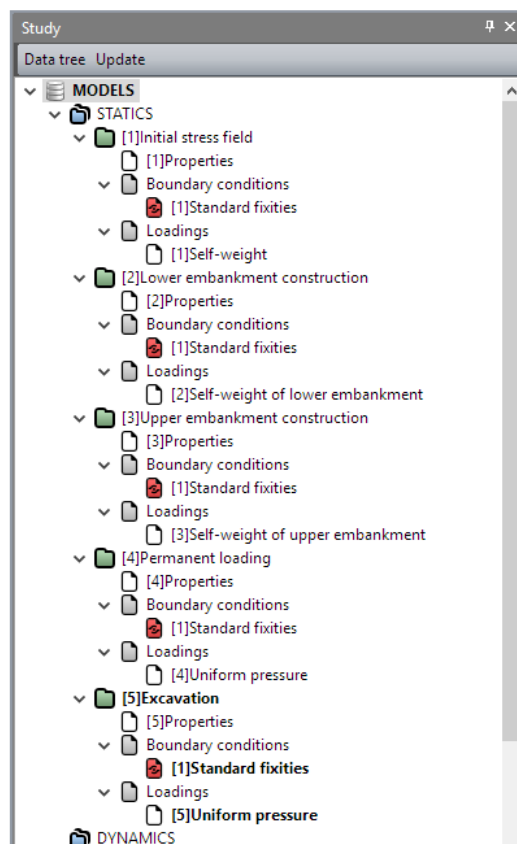
Figure 14 : View of the STUDY tree after Excavation model definition

Active/inactive bodies

The excavation operation leads to set as inactive the related volume body.

1. Select the volume body representing the excavation.

2. Click on *Activate/deactivate bodies*.

3. In the grid, tick "Inactive".

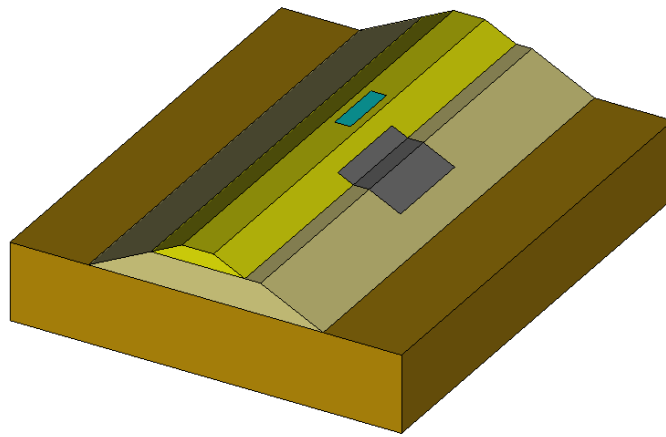4. *Apply*. The volume body is now marked with the neutral colour.

Figure 15 : View of the model after inactivation of the excavated body

Boundary conditions:

There is no change in the boundary conditions.

Loading set:

1. In the tree, rename the default loading set (*LoadSet1*) using [F2] into **Excavation forces**.
2. Activate the *LOADS* tab.
3. On the toolbar, activate *Excavation forces*.
    -   Tick "Active"
    -   Set the value of lambda to 1,
    -   As stress origin, select **Permanent loading** model as stress field origin.
    -   The button **Automatic selection** is active. Click on it, it will automatically select the faces released by the inactivation of groups.
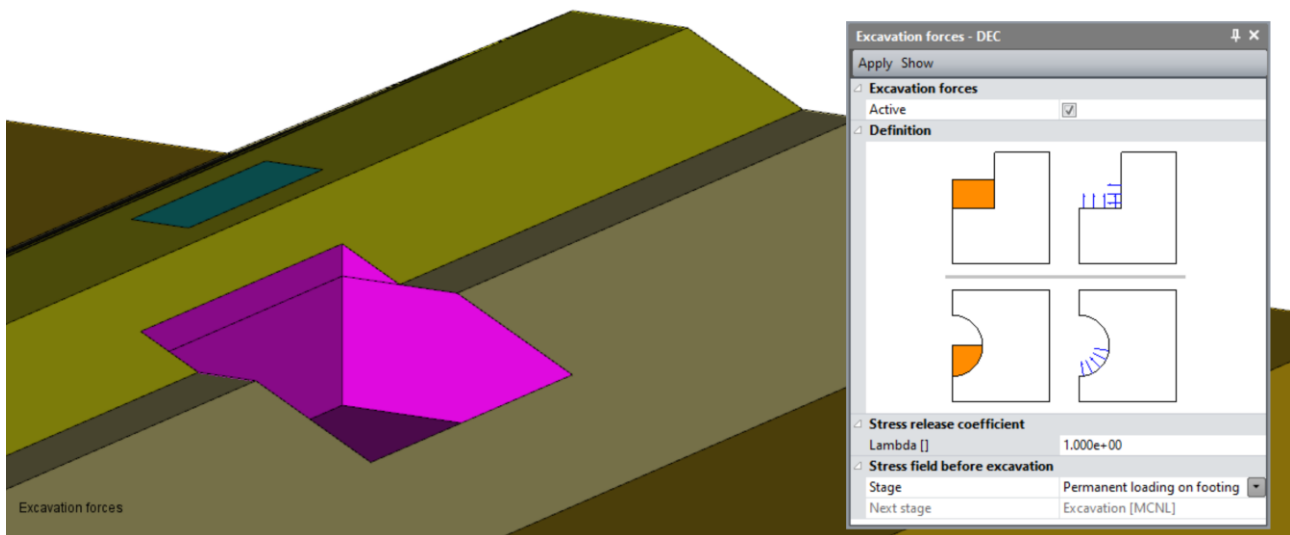4. Click on *Apply*. The excavation forces are displayed.

Figure 16 : View of the excavation forces

Calculation parameters:

No modifications from previous model.

4. SOLVE

We launch the calculations simultaneously. It is obviously possible to launch the calculations one by one.

1. Go to the *ANALYSIS* tab.

2. Click on [icon] *Analysis manager*.
    - Select all the models using the ☑ tool.
    - Select Create input files for the solver and calculate. Click on Validate.

3. The iteration process is displayed on the **Working window**. It ends with the message "End of analysis in EXEC mode".

⚠ CESAR-LCPC detects if the models are ready for calculation. All steps should be validated with a tick mark.

⚠ The calculations duration depends of the configuration of the computer. As example, for a labtop with 16 Go, each stage is calculated in few seconds (10 to 20").
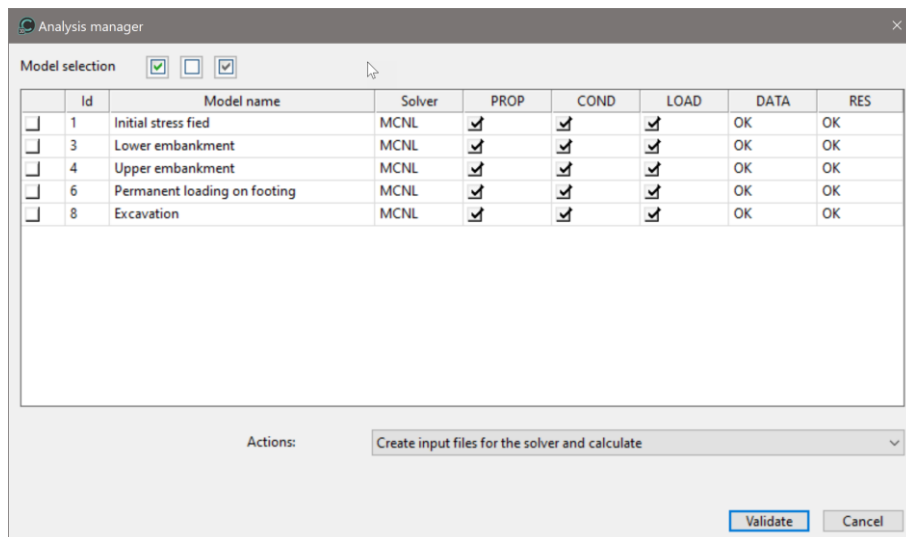
💡 All the messages during the analysis will be shown in an **Output Window**. Especially, one needs to be very cautious about warning messages, because these messages indicate that the analysis results may not be correct. The result is saved as a binary file (*.RSV4) in the temporary folder (.../TMP/), defined during setup. The detailed analysis information is also saved in a text file (*.LIST).

5. RESULTS ANALYSIS

We display the deformed model and scalar plots of total displacement for the last 2 stages.

1. Go to the tab *RESULTS*.
2. Il the list of models, select **Permanent loading**.
3. Activate **[icon]** *Result Types*
   - Select **Deformed** as Mesh,
   - Tick **Contour plots**, in the list select **Total displacement**,
   - Click on *Apply*.
4. Activate **[icon]** *Mesh settings*.
   - Select **Group borders**, as mesh border style,
   - *Apply*.
5. Activate **[icon]** *Contour plots settings*.
   - Tick **Contouring**, as isovalues style,
   - Tick **Contour lines** and select **Grey** colour,
   - *Apply*.
6. Activate **[icon]** *Displacement settings*.
   - Select **Manual** as scalar type,
   - Set a value of **10 mm** of displacement represented by **0,5 m** (of the model size),
   - *Apply*.
7. Activate **[icon]** *Legend*.
   - Select **Contour plot** as *Legend type*,
   - Activate *Legend border*,
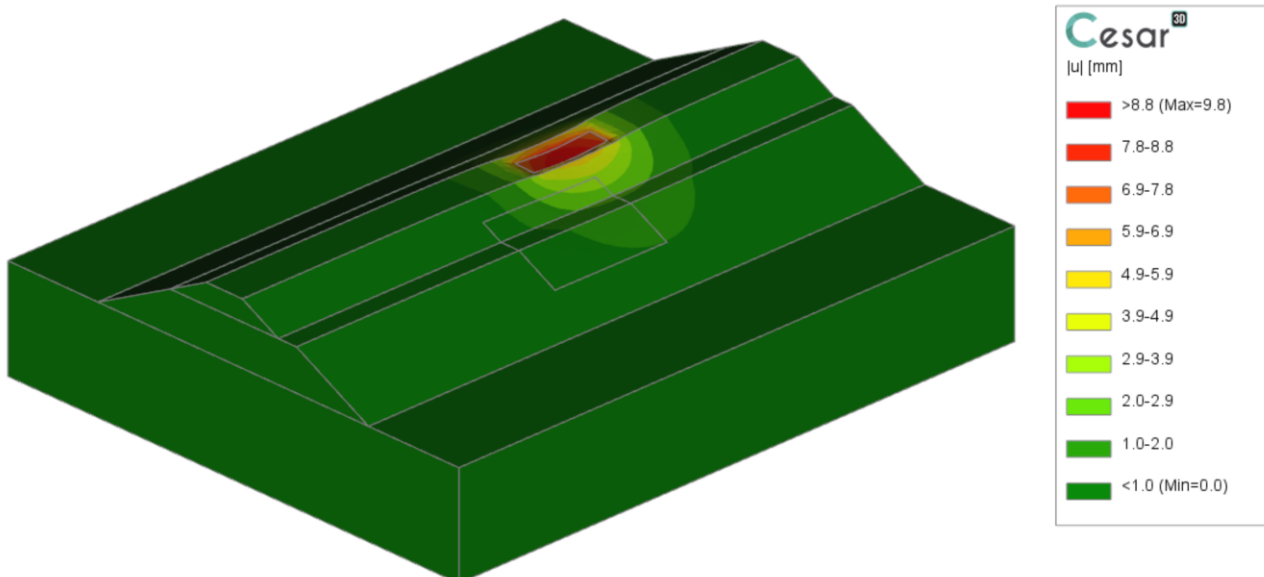   - Activate *Background colour*,
   - *Apply*.

Figure 17: Contour plots of total displacements under permanent loading.

1. Select the model **Excavation**.
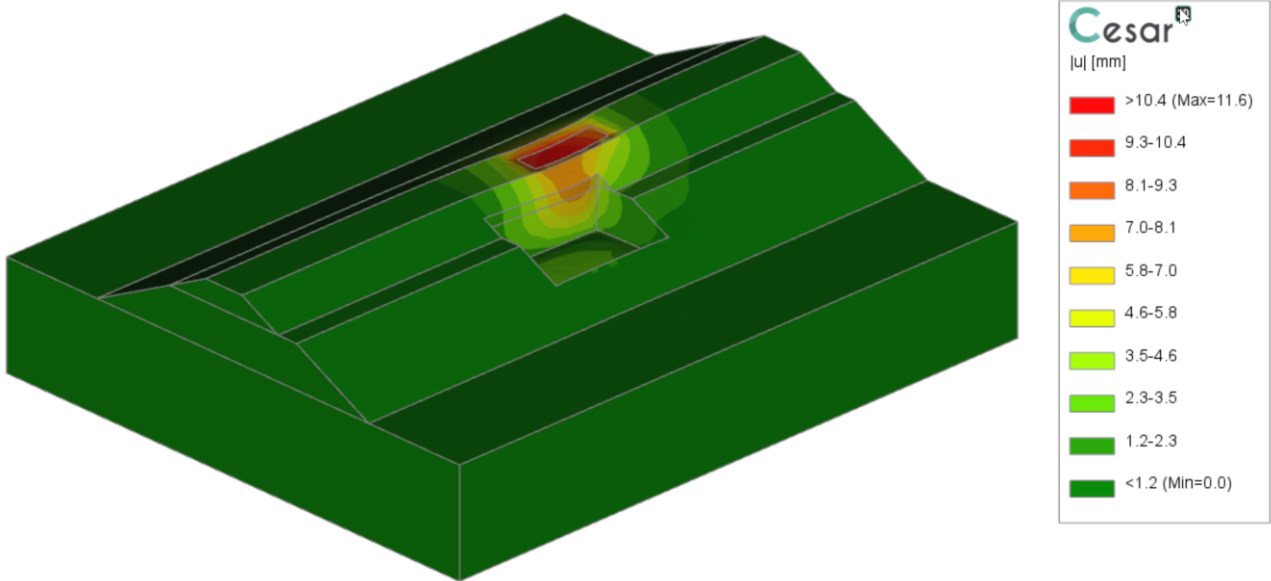2. Display of results is automatically updated.

Figure 18: Contour plots of total displacement after excavation.

3. Activate ☑/ *Displacement settings*.
   - Activate **Incremental displacement**,
   - Select the stage "Permanent loading" as reference model,
   - *Apply*.

Doing so, we display the incremental displacements associated to the excavation forces.
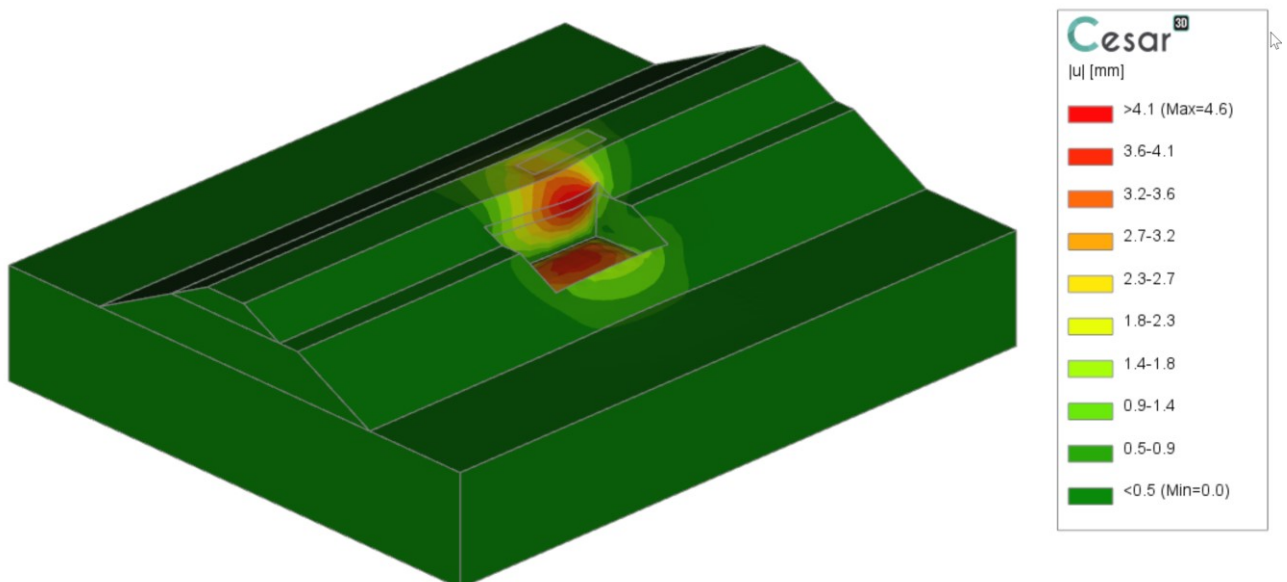

Figure 19: Contour plots of incremental total displacement after excavation.

Edited by:

8 quai Bir Hakeim

F-94410 SAINT-MAURICE

Tel.: +33 1 49 76 12 59

cesar-lcpc@itech-soft.com

www.cesar-lcpc.com